(NASA-CR-161321) USER'S GUIDE: SPAR PROCESSOR MN ANALYSIS OF INELASTIC THREE-DIMENSIONAL SOLIDS, PART 2 (Engineering Information Systems, Inc.) 16 p HC A02/MF A01 CSCL 20K G3/39

N80-10514

Unclas
45904

EISI ENGINEERING INFORMATION SYSTEMS, INC.

5120 CAMPBELL AVENUE, SUITE 240
SAN JOSE, CALIFORNIA 95130

EISI/A3088

USER'S GUIDE: SPAR PROCESSOR MN

ANALYSIS OF INELASTIC THREE-DIMENSIONAL SOLIDS

NAS8-36365 (Final Report - Part 2)

by

W. D. Whetstone and C. E. Jones

September 1979

ENGINEERING INFORMATION SYSTEMS, INC.
5120 CAMPBELL AVENUE, SUITE 240
SAN JOSE, CALIFORNIA 95130

PREFACE

This report was prepared by Engineering Information Systems, Inc., for the George C. Marshall Space Flight Center of the National Aeronautics and Space Administration. Together with Revision 3 to EISI/A2200, this report constitutes the final report for NASA-MSFC Contract NAS8-32365.

Submitted by:

Engineering Information Systems, Inc.

W. D. Whetstone
President

CONTENTS

Section 1

INTRODUCTION AND MN COMMAND SUMMARY

Processor MN may be used, in conjunction with other processors, to analyze systems in which some or all of the three-dimensional elements (S41, S61, S81) have nonlinear stress-strain relations. In the current implementation (9/79), the following representations of nonlinear material behavior are provided:

1- Von Mises yield criterion and Prandtl-Reuss flow rule, given a piecewise linear uniaxial stress-strain curve; assuming monotonic loading, i.e. unique stress-strain relation, regardless of the state history.

2- The "mechanical sublayer" method, in which the material is assumed to consist of a number of "layers" equal to the number of linear segments on the uniaxial stress-strain curve. Each layer is assumed to be elastic/ perfect plastic.

Method 1 generally applies only if loading of all elements is progressively increased, with no unloading of yielded elements. Method 2 involves greater execution costs and more mass data storage, but does furnish a more appropriate representation of inelastic behavior during unloading, e.g. the Bauschinger effect.

The user defines the properties of each material by creating, through AUS/TABLE, a data set containing (a) a numeric code - either 1. or 2. - indicating which of the above representations is to be used, (b) Poissons ratio, for the elastic range, and (c) stress-strain values defining points on a uniaxial stress-strain curve. Any number of materials may be used, and each stress-strain curve may have any number of linear segments. If the mechanical sublayer method is used, the number of linear segments - hence the number of layers - usually should not be more than 4 to 10, since execution costs and data storage requirements are significantly affected by the number of layers. The system may also contain linear one and two-dimensional elements, e.g.E21, E43.

The primary data sets used to represent the state of the system are:

- The total nodal displacements, normally residing in a SYSVEC-format data set named TOTAL DISPLACEMENTS, and

- The element stress state, residing in data sets named NLSS Sij, where Sij is S41,S61, and/or S81. The content of the stress state arrays is discussed in Section 4.

Command	Function
TRMC	Transform Material Constants. This command is normally given only one time, at the beginning of a solution process. The result is the production, for each material (m), of a data set named NONL CONS m, based on user-supplied data contained in data set STRESS CURVE m. The required content of the STRESS CURVE data sets is discussed in Sections 4 and 5.
INSS	Initialize Stress State. This command causes the stress state arrays to be initialized, within data sets named NLSS Sij. Sij is S41, S61, and/or S81, depending on the type of elements present. The stress state arrays will subsequently be updated (overwritten), by the NF sub-function described subsequently.
NF	Nodal Force and stress state computation. The primary source data sets required are (1) a data set named STAT DISP defining an increment to the nodal displacements, and (2) the stress state at the beginning of the increment, residing in the NLSS Sij data sets. The outputs are: (1) A SYSVEC format data set named NF, containing the nodal forces which are necessary for equilibrium in the new nodal displacement state. If data set NF already exists, it will be overwritten with the newly computed nodal forces. (2) The NLSS Sij data sets are overwritten with the newly-computed stress state.
TK	Tangent K computation. This command causes the tangent elastic stiffness matrices for all S41, S61, and S81 elements to be computed, based on the current stress state contained in the NLSS Sij data sets. The tangent K's are stored in the Sij EFIL data sets, overwriting the intrinsic k's previously computed by either EKS or MN/TK.
PRINT	This command causes the stress state to be printed for all S41, S61, and S81 elements. If the command PRINT,1 is given, the display will include the stresses in individual layers, if the mechanical sublayer representation is used. Stresses at the 8 Gauss quadrature points are displayed.

Section 2

TYPICAL RUNSTREAM FOR NONLINEAR ANALYSIS

To illustrate typical uses of MN commands TRMC, INSS, NF, TK and PRINT in the context of an overall nonlinear analysis, the following example of an elementary Newton procedure is shown. More general procedures, e.g. variations of Newton-Raphson may be designed by the user to suit the needs of particular classes of problems.

In this example, it is assumed that the load is applied in a succession of steps. Following the initial problem set-up, the basic task is: given the loads currently acting (CAF), find a system state such that the corresponding nodal forces required for equilibrium (NF, as computed by MN) are essentially equal to CAF.

Initial
set-up:

```
*XQT TAB                                Create a complete finite element model,
 - -                                    as though a linear analysis was to be
 - -                                    performed.
*XQT AUS
 TABLE(NI=31,NJ= -)
 PROP BTAB 2 21
 - -
*XQT ELD
 - - NSECT= material number$            NSECT points both to a line in the
 - -                                    PROP BTAB 2 21 data set, and to the
 - - NSECT= material number             STRESS CURVE nsect data set.
*XQT E
*XQT EKS
*XQT TOPO
*XQT K
*XQT INV

 Compute linear static solutions
 as necessary to validate the
 basic model, using AUS/SYSVEC,
 SSOL, etc.


*XQT AUS
 table(NI= -): STRESS CURVE 1$          Stress curve, etc., for material 1
 code, nu,   s1,e1  s2,e2, - -
 TABLE(NI= -): STRESS CURVE 2$          Stress curve, etc., for material 2
 code, nu,   s1,e1  s2,e2, - -
 - -
 - -

 SYSVEC: TOTAL DISPLACEMENTS$             Initialize to zero.
        I=1: J=1: .0
 SYSVEC: NF FORC$                         Initialize to zero the current nodal
        I=1: J=1: .0$                     force array to be overwritten by MN/NF
```

For each
load step:

	*XQT AUS	
	SYSVEC: CAF$	Define current active forces.
	- - -	
	- - -	
STEP-A	*XQT AUS	
	APPLIED FORCES=SUM(CAF, -1. NF)$ =	Difference between the current active forces, and the nodal forces most recently computed by MN/NF.
		GO TO STEP-B IF APPL FORC IS ESSENTIALLY ZERO.
	*XQT K	
	*XQT INV	
	*XQT SSOL	Compute STAT DISP corresponding to APPL FORC.
	*XQT MN	
	NF$	Compute new nodal forces, NF, and current stress state, corresponding to the change in displacements, STAT DISP.
	TK$	Compute element tangent K's corresponding to current stress state.
	*XQT AUS	
	DEFINE TD= TOTAL DISPLACEMENTS	
	TOTAL DISP=SUM(TD, STAT)	New total displacements = previous total + STAT DISP.
	RETURN TO STEP-A	
STEP-B	*XQT MN	Convergence achieved.
	PRINT	Print stresses.
	*XQT VPRT	
	PRINT TOTAL DISPLACEMENTS	Print total joint motions.

Section 3

RESET CONTROLS, CENTRAL MEMORY REQUIREMENTS, AND ERROR CODES

3.1 RESET Controls

Name	Default Value	Meaning
CLIB	1	Material constant library, containing all STRESS CURVE nmat and NONL CONS nmat data sets.
ELIB	1	Library containing all Sij EFIL data sets.
NSET	1	Value of nset in data set names indicated below.
NCON	1	Value of ncon in data set names indicated below.
SLIB	1	Library containing all NLSS Sij nset ncon data sets.
ULIB	1	Library containing STAT DISP nset ncon.
FLIB	1	Library containing NF FORC nset ncon.
ZDE	1.-20	Zero divide test, (delta stress)/(delta strain)- stress curves.
LAYERS	5	Maximum number of "layers" in any material, if the mechanical sublayer method is used. Sets the size of the stress state arrays, NLSS Sij nset ncon.
NJSS	10	Sets the number of elements per block in the NLSS Sij arrays.
zyield	.001	Test for yielding: (effective stress - effective stress at yield)/ (effective stress at yield) .lt. zyield.

3.2 Central Memory Requirements.

To execute the NF command, data space must be sufficient to simultaneously contain all of the following:

- All of the NONL CONS data sets, typically about 50 words per material.
- One block of an NLSS Sij data set. The block size of these data sets is controlled by RESETs LAYERS and NJSS. As indicated in Section 4, the block length is 48 x "layers" x "njss".
- 12 times the number of system joints.

The other commands, e.g. TK, require approximately the same data space as NF, less 12 times the number of system joints.

3.3 Error Codes.

The following error codes originate in MN:

NERR	NIND	Meaning
MN	1	Source NLSS Sij data set in error.
MN	2	Insufficient data space, operation NF, TK, or PRINT
MN	10000	Unknown command.
INSS	1	Insufficient data space, command INSS.
TRMC	2	Source STRESS CURVE data set improperly dimensioned.
TRMC	3	Insufficient data space.
TRMC	4	Poissons ratio greater than 0.5.
TRMC	5	Infinite modulus of elasticity.
TRMC	6	Illegal material code = first word of STRESS CURVE.
TRMC	7	Initial modulus =0. An error only if mechanical sublayer used.
CONS	1	NONL CONS data set in error.
CONS	3	No NONL CONS data sets present in CLIB.
NF3D	1	Required NONL CONS data set not present, operation NF.
NF3D	2	A material has more layers than prescribed by RESET LAYERS.
TK3D	1	Same as NF3D 1, except during TK operation.
TK3D	2	Same as NF3D 2, except during TK operation.
PRNL	1	Same as NF3D 1, except during PRINT operation.
PRNL	2	Same as NF3D 2, except during PRINT operation.

Section 4

DATA SET CONTENTS

In the following, m indicates a material number, n is the number of straight line segments on a uniaxial stress-strain curve, and nmax is the greatest number of layers of any material, as prescribed through the LAYERS reset control.

Name	NI	NJ	Contents
STRESS CURVE m 0	(2+2n)	1	Material type code, Poissons ratio, s1,e1, s2, e2, - - sn,en. The type code is 1.0 for method 1, i.e. unique stress-strain curve assuming monotonic loading; and 2.0 for the mechanical sublayer method.
NONL CONS m 0			Created by MN/TRMC, based on data contained in STRESS CURVE m. The content varies with material type code. Contains information such as layer yield stresses and participation factors.
NLSS Sij nset ncon	48*nmax	njss	Nset, ncon, nmax, and njss all are determined by RESET controls. For each element, 6 stress components at each of 8 Gauss quadrature points are stored, for each "layer." If the mechanical sublayer method is not used for any material, LAYERS should be RESET to 1, since only the total stress is stored.

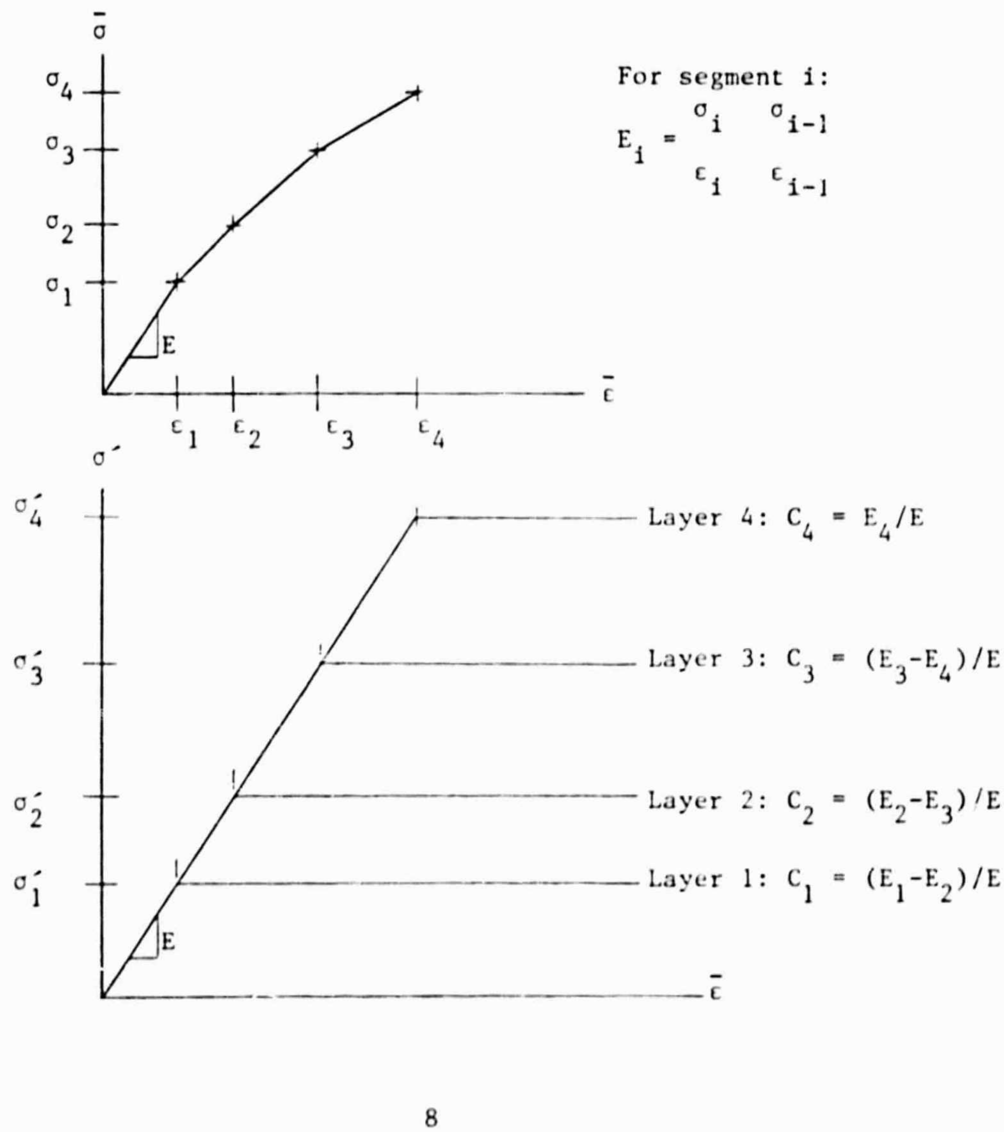
Section 5

THEORY

5.1 Yield Criteria, Flow Rules, and Constitutive Relations.

Details of generally used yield criteria, flow rules, and constitutive relations are given in the references listed in Section 5.4. The mechanical sublayer method, indicated by type code = 2.0 in the STRESS CURVE data sets, is described briefly below. See references 1 and 2, Section 5.4, for more details.

The mechanical sublayer method assumes that a given material, idealized as having a piecewise-linear uniaxial stress-strain curve, consists of a number of coincident components or layers, one for each straight line segment on the stress-strain curve. Each layer is elastic - perfect plastic. All layers have the same modulus of elasticity and Poissons ratio, but each layer has a unique yield stress, determined as indicated on the figure shown below. The Ci's are the layer participation, or weighting factors; i.e. the Ci's total 1.0. For each individual layer, the Von Mises yield criterion and Prandtl-Reuss flow rule are used to determine incremental stress-strain coefficient matrices, for use in computing changes in element stress states and tangent elastic stiffness matrices.



5.2 Nodal Force Computations - NF Command.

The method used to compute the nodal forces necessary for equilibrium following an increment to the nodal dislacements is as follows:

- The change in strain corresponding to the change in displacements is determined on the basis of an assumed linear isoparametric displacement field.

- The corresponding new stress state in each layer, at each of the 8 Gauss quadrature points is computed, and a new material constitutive matrix is determined for each quadrature point.

- The nodal forces are computed, corresponding to the virtual work associated with a linear isoparametric displacement perturbation of the element.

5.3 Tangent K Computation - TK Command.

The tangent K computations are performed exactly the same as for the linear S41, S61, and S81 hybrid elements, except that in computing the internal strain energies, the current values of the material compliance matrices at the Gauss quadrature points are used, rather than the initial elastic constants.

5.4 References.

1. Spilker, R. L. "Elastic-Plastic Analysis by the Hybrid-Stress Model and the Initial Stress Approach," Technical Report AMMRC CTR-73-38 and MIT ASRL TR 172-1 (Dept. of Aeronautics and Astronautics, MIT).

2. Nayak, G. C., and O. C. Zienkiewicz. "Elasto-Plastic Stress Analysis. A Generalization for Various Constitutive Relations Including Strain Softening," Int. J. Num. Methods in Engineering, Vol. 5 (1972), 113-135.

3. Cook, Robert D. "Concepts and Applications of Finite Element Analysis." Wiley, New York, 1974. 402 pp.

4. Luk, C.H. "Assumed Stress Hybrid Finite-Element Method for Fracture Mechanics and Elastic-Plastic Analysis," Technical Report AFOSR TR 73-0493, and MIT ASRL TR 170-1 (1972) (Dept. of Aeronautics and Astronautics, MIT).

5. Pian, T. T. H. "Application of Hybrid Finite Elements to Structural Mechanics Problems," Technical Report AFOSR TR 77 6209 (Aeroelastic and Structures Research Laboratory, MIT).

6. Pian, T. T. H. "Variational Principles for Incremental Finite Element Methods," J. Franklin Inst., Vol. 302, Nos. 5 and 6, Nov. and Dec. 1976.

7. Pian, T. T. H., P. Tong, C. H. Luk, and R. L. Spilker. "Elastic-Plastic Analysis by Assumed Stress Hybrid Model," Proceedings of the 1974 International Conference on Finite Element Methods in Engineering, J. A. Pulamo and A. P. Kabaila, eds. (1974). 419-434.

8. Scharnhorst, T., and T. T. H. Pian. "Finite Element Analysis of Rubber-Like Materials by a Mixed Model," Int. J. Num. Methods in Engineering, Vol. 12 (1978), 665-676.

9. Yamada, Y. S., and K. Takatsuka. "Elastic-Plastic Analysis of Saint-Venant Torsion Problem by a Hybrid Stress Model," Int. J. Num. Methods in Engineering, Vol. 5 (1972), 193-207.

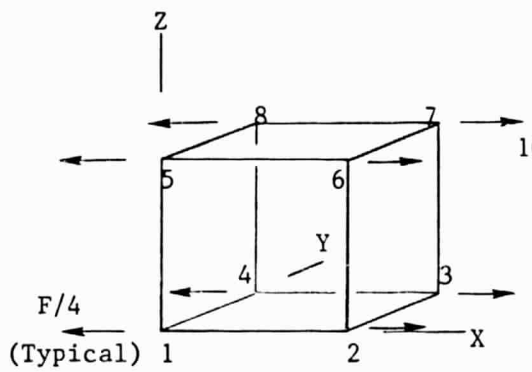
10. Zienkiewicz, O. C. "The Finite Element Method in Engineering Science," McGraw-Hill, London, 1971. 521 pp.

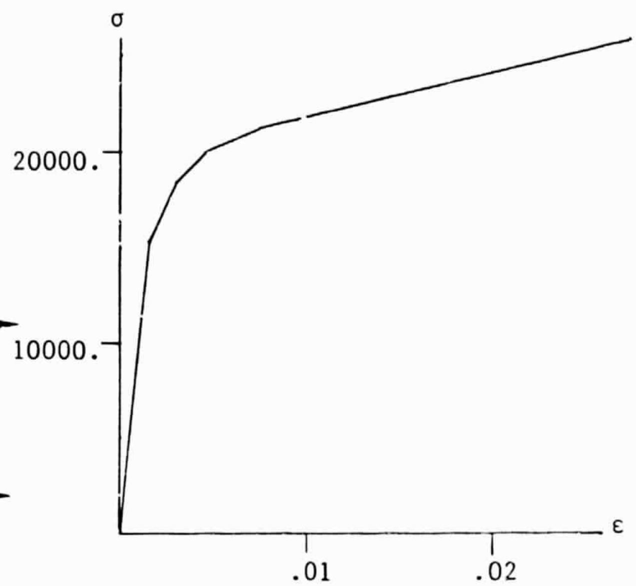
11. Zienkiewicz, O. C., S. Valliappan, and I. P. King. "Elasto-Plastic Solutions of Engineering Problems 'Initial Stress,' Finite Element Approach," Int. J. Num. Methods in Engineering, Vol. 1 (1969), 75-100.

Section 6

EXAMPLE

The runstreams shown on the following page were used to solve the problem described below. The element is cubic, with edge length = 1.0. Uniform axial loading along the X axis is applied, and the extension computed.





The stress-strain curve is defined by the following points:

Stress	Strain
15300.	.00153
18500.	.00300
20000.	.00450
21300.	.00750
26900.	.02650

The computed results, which agree exactly with the stress-strain curve are:

Step	F	Displacement
1	14000.	.0014
2	16300.	.00199
3	19250.	.00375
4	21000.	.00681
5	22750.	.01242
6	25000.	.02000

Initial problem set-up:

```
@XQT TAB
 START         8 4 5 6
 MATC:         1 .1+8 0.
 JLOC:         1  0.  0.  0.  1.  0.  0. 2 1
               3  1.  1.  0.  0.  1.  0. 2 1
               5  0.  0.  1.  1.  0.  1. 2 1
               7  1.  1.  1.  0.  1.  1. 2 1
 CON=1:        ZERO 1: 1: 4: 5: 8: ZERO 2: 1: ZERO 3: 1: 4
@XQT AUS
 TABLE(NI=31,NJ=1): PROP BTAB 2 21: J=1: 1.>
  1.-7>
  -.3-7 1.-7>
  -.3-7 -.3-7 1.-7>
  0.   0.   0.   2.6-7>
  0.   0.   0.   0.   2.6-7>
  0.   0.   0.   0.   0.   2.6-7>
  0. 0. 0. 0. 0. 0. 0. 0. 0.
@XQT ELD
 S81:          1 2 3 4 5 6 7 8
@XQT E
@XQT TOPO
@XQT AUS
 TABLE(NI=12,NJ=1): STRESS CURVE: I=1 2: J=1: 1. .333333
 I=3 5 7  9 11: J=1: 15.3+3  18.5+3  20.0+3  21.3+3  26.9+3
 I=4 6 8 10 12: J=1: .00153  .00300  .00450  .00750  .02650
@XQT MN
 RESET LAYERS=1: TRMC: INSS: TK
@XQT AUS
 SYSVEC: NF FORCE: I=1: J=1: 0.: TOTAL DISP=UNION(NF)
```

Typical iterative sequence. Load step 2 is shown:

```
@XQT AUS
 SYSVEC: CAF: I=1: J=2: 3: 6: 7: 4.075+3: APPL FORC=SUM(CAF, -1. NF)
@XQT K
@XQT INV
@XQT SSOL
@XQT MN
 RESET LAYERS=1: NF: TK
@XQT AUS
 DEFINE STAT=STAT DISP: DEFINE TD=TOTAL DISP: TOTAL=SUM(TD,STAT)
 APPLIED FORCES=SUM(CAF, -1. NF)
@XQT K
@XQT INV
@XQT SSOL
@XQT AUS
 DEFINE STAT=STAT DISP: DEFINE TD=TOTAL DISP: TOTAL=SUM(TD,STAT)
@XQT VPRT
 PRINT TOTAL'TOTAL DISPLACEMENTS, CAF=16300
@XQT MN
 RESET LAYERS=1: NF: PRINT
```